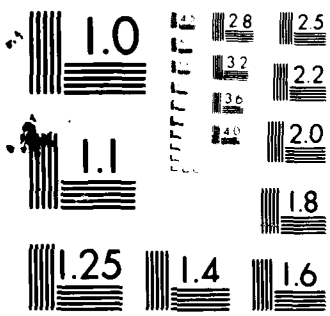
AD-A107 095 DAYTON UNIV OH RESEARCH INST F/G 13/13
FINITE ELEMENT TECHNIQUES FOR STATIC STRESS AND FREE VIBRATION --ETC(U)
MAR 81 R A BROCKMAN F33615-77-C-3075
UNCLASSIFIED UDR-TR-80-134 AFWAL-TR-81-3007 NL

1 of 1

END
DATE
FILMED
12-81
DTIC




MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

AFWAL-TR-81-3007



# FINITE ELEMENT TECHNIQUES FOR STATIC STRESS AND FREE VIBRATION ANALYSIS OF SANDWICH COMPOSITES

R. A. Brockman

University of Dayton
Research Institute
Dayton, Ohio 45469

March 1981



Final Report for Period September 1977 - November 1980

Approved for public release; distribution unlimited.



FLIGHT DYNAMICS LABORATORY
AIR FORCE WRIGHT AERONAUTICAL LABORATORIES
AIR FORCE SYSTEMS COMMAND
WRIGHT-PATTERSON AIR FORCE BASE, OHIO 45433

NOTICE



This technical report has been reviewed and is approved for publication.

Harold C. Croop

HAROLD C. CROOP, AFWAL/FIBC
Project Engineer

Larry G. Kelly

LARRY G. KELLY, Chief,
Structural Concepts Branch

FOR THE COMMANDER

Olsen

JAMES J. OLSEN, Assistant for
Research & Technology
Structures and Dynamics Divis on

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

| REPORT DOCUMENTATION PAGE | | READ INSTRUCTIONS BEFORE COMPLETING FORM |
|---|---|---|
| 1. REPORT NUMBER<br>AFWAL-TR-81-3007 | 2. GOVT ACCESSION NO.<br>AD-A107095 | 3. RECIPIENT'S CATALOG NUMBER |
| 4. TITLE (and Subtitle)<br>FINITE ELEMENT TECHNIQUES FOR STATIC STRESS AND FREE VIBRATION ANALYSIS OF SANDWICH COMPOSITES. | | 5. TYPE OF REPORT & PERIOD COVERED<br>Final Rep[redacted]t [redacted] Sep[redacted]77 - Nov[redacted]80 |
| | | 6. PERFORMING ORG. REPORT NUMBER<br>UDR-TR-80-134 |
| 7. AUTHOR(s)<br>Robert A./Brockman | | 8. CONTRACT OR GRANT NUMBER(s)<br>F33615-77-C-3075 |
| 9. PERFORMING ORGANIZATION NAME AND ADDRESS<br>University of Dayton Research Institute<br>300 College Park Avenue<br>Dayton, Ohio 45469 | | 10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS<br>62201F<br>2401/03/11 |
| 11. CONTROLLING OFFICE NAME AND ADDRESS<br>Air Force Wright Aeronautical Laboratories<br>AFWAL/FIBCB<br>Wright-Patterson Air Force Base, OH 45433 | | 12. REPORT DATE<br>Mar[redacted]81 |
| | | 13. NUMBER OF PAGES<br>61 |
| 14. MONITORING AGENCY NAME & ADDRESS(if different from Controlling Office) | | 15. SECURITY CLASS. (of this report)<br>Unclassified |
| | | 15a. DECLASSIFICATION/DOWNGRADING SCHEDULE |

16. DISTRIBUTION STATEMENT (of this Report)

Approved for public release; distribution unlimited.

17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)

18. SUPPLEMENTARY NOTES

19. KEY WORDS (Continue on reverse side if necessary and identify by block number)

Sandwich Composites
Finite Element Method
Stiffened Panels
Vibration Analysis
Thin Shells
Thermal Stress

20. ABSTRACT (Continue on reverse side if necessary and identify by block number)

Finite element solution techniques for stress and vibration analysis of layered sandwich panels are presented. The modeling approach considers each layer of the sandwich explicitly and can treat panels involving discrete stiffeners of two types. Static analyses of general sandwich panels can include mechanical and/or thermally-induced forces. Free vibration analysis is also considered. The associated computer program is suitable for

DD FORM 1 JAN 73 1473 EDITION OF 1 NOV 65 IS OBSOLETE

Unclassified

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

20. (continued)

operation at an interactive terminal, and includes provisions for automatic mesh generation and tabular output. Generated input and output data are in a form which may be accessed by existing computer graphics programs for plotting geometry and results. Sample analyses are presented to demonstrate the developed techniques.

## FOREWORD

This report describes work performed by the University of Dayton Research Institute (UDRI) under Air Force Contract F33615-77-C-3075, Structural Sandwich Composites. The effort was conducted for the Flight Dynamics Laboratory, Air Force Wright Aeronautical Laboratories, under the administration and technical direction of the Air Force Project Engineer, Mr. Harold C. Croop (AFWAL/FIBCB).

Administrative project supervision at the UDRI was provided by Mr. Dale H. Whitford (Supervisor, Aerospace Mechanics Division), and technical supervision was provided by Dr. Fred K. Bogner (Group Leader, Analytical Mechanics Group).

# TABLE OF CONTENTS

LIST OF ILLUSTRATIONS

## SECTION 1
## INTRODUCTION

Structural sandwich composites are commonly employed in the design of structural components requiring high stiffness-to-weight ratio characteristics. Such materials are particularly attractive for aerospace applications in which high performance and maximum payload are typically sought.

While sandwich materials offer attractive structural characteristics for many applications, effective design of sandwich construction is generally difficult due to the complexity of response and the possibility of failure in modes which are unique to sandwich materials. In order to make effective use of these advanced material concepts, improved analytical techniques are needed for use in design applications. The scope of analysis must be broad, including static and dynamic response for most applications, buckling analysis for sandwich panels and shells, and thermal stress calculations for heat-shielding and similar components.

The development presented in this report focuses upon such problems as static response, thermal stress determination and natural frequency calculations, all of which may be repeated a number of times during preliminary design and are thus most useful when analyzed in an interactive mode. Buckling, large deflection response and plastic analysis of sandwich often require a rather detailed and time-consuming analysis; such nonlinear problems are discussed in a companion report[1]. Similar interactive techniques for buckling analysis and design optimization are also available[2,3].

### 1.1 GENERAL APPROACH

The analytical techniques reported here are oriented toward layered sandwich panels, including stiffening elements and edge members, and are limited to linear elastic responses. Due to the practical requirements of considering many different types of boundary conditions and stiffener arrangements, and possibly of considering non-rectangular or curved panel geometries, the finite element method is chosen as the analytical basis throughout. With the finite element approach, only the task of data generation need be specialized to particular classes of geometries for interactive analysis.

The particular finite elements employed are derived from the same principles as the nonlinear elements developed in Reference 1. A typical sandwich panel is discretized using thin shell-type elements for the face sheet layers, and three-dimensional, shear-flexible elements in the core layer. The shell (face sheet) elements are formulated using a penalty function approach which eliminates the need for slope compatibility between elements and permits the joining of shell (face sheet) and solid (core) elements directly in a straightforward manner. Stiffening members derived from an approach similar to the shell elements can be used to model either face sheet stiffeners or full-depth spars and edge closeouts. In each case, displacement compatibility is achieved with the more complex elements used to represent the layers of the sandwich panel itself. Consistent static and thermal loadings are provided for each of these classes of elements in static analysis, as well as consistent mass formulations for natural frequency calculation.

## 1.2 SCOPE OF ANALYSIS

In the present development, the linear (small displacement, elastic) behavior of sandwich materials is considered for the cases of static loading and free vibration. For the case of static analysis, both mechanical and thermally-induced forces may be treated. Provisions are made for performing analyses of models of moderate size; when larger analyses are necessary, the data generated in the interactive mode may be used in conjunction with the larger finite element program described in Reference 1.

Due to limitations in the data generation phase of the computer program, the class of sandwich panels considered is flat and rectangular, with arbitrary boundary conditions at each edge. However, the data generation scheme is organized so that geometries for non-rectangular or curved panels may be generated through minor modification of the existing code (see Section 4). Furthermore, the input information accepted by the analysis program is in the formats described in Reference 1, so that data also may be generated manually and the use of existing interactive plotting software[4] is possible.

Each layer of a sandwich panel is modeled explicitly in the present analysis, so that localized deformations and thermal stresses due to non-uniform heating can be represented with high accuracy. The use of a layered model also facilitates the representation of face sheet and full-depth stiffeners or panel edge members. These stiffening elements may be located arbitrarily within the sandwich panel.

## SECTION 2
## THEORETICAL FORMULATION

The theoretical basis of each class of finite element (face sheet, core, stiffener) used in the present development is described in the following paragraphs. Further details of the element formulations may be found in References 2, 5 and 6. In each case the starting point is the element potential energy $\Pi_p$; for static problems, the true state of equilibrium is defined simply by the condition

$$\delta\Pi_p = 0. \tag{1}$$

In dynamic (natural frequency) analysis Hamilton's principle

$$\delta\int_{t_1}^{t_2}(T - \Pi_p)dV = 0 \tag{2}$$

provides the governing equations, in which T is the kinetic energy.

### 2.1 THIN SHELL/FACE SHEET ELEMENTS

The development of shell-type finite elements suitable for the representation of sandwich face sheets is complicated by the compatibility requirements for elements derived from classical thin plate or shell theories. The rotational degrees of freedom dictated by the condition of slope continuity preclude the direct enforcement of compatibility with standard isoparametric solid elements. If the layers of the sandwich are to be modeled explicitly, it is necessary either to derive plate/shell elements using only displacement degrees of freedom at the bond lines or to construct higher-order solid elements to represent the core layers. In the present work, the former approach is adopted. The thin plate or shell approximation is obtained herein by appropriate specialization of three-dimensional continuum theory, as described in References 5 and 6.

The undeformed geometry of a typical element of a shell is pictured in Figure 1. Local coordinates x,y are imbedded in the base plane of the element, and the height of the middle surface above the base plane is denoted by $Z(x,y)$. Distances away from the midsurface are measured by a third independent coordinate, $\zeta$. The position of a generic point in the shell is then

$$\vec{r} = x\hat{i} + y\hat{j} + z\hat{k} + \zeta\hat{n} \tag{3}$$

where, if the shell is locally shallow, the unit normal vector $\hat{n}$ is

$$\hat{n} = -z,_x\hat{i} - z,_y\hat{j} + \hat{k}. \tag{4}$$

The assumption of local shallowness also implies that the coordinate system $(x,y,\zeta)$ is approximately Cartesian.

If the same element of shell material is subject to a displacement field

$$\vec{u} = u\hat{i} + v\hat{j} + w\hat{k}, \tag{5}$$

the displaced position of an arbitrary point is

$$\vec{R} = (x - \zeta z,_x + u)\hat{i} + (y - \zeta z,_y + v)\hat{j}$$

$$+ (z + \zeta + w)\hat{k} \ . \tag{6}$$

By comparing Equations 3 and 6, the three dimensional strains can be identified, since

$$2e_{ij}dx_i dx_j = \vec{d}R \cdot \vec{d}R - \vec{d}r \cdot \vec{d}r. \tag{7}$$

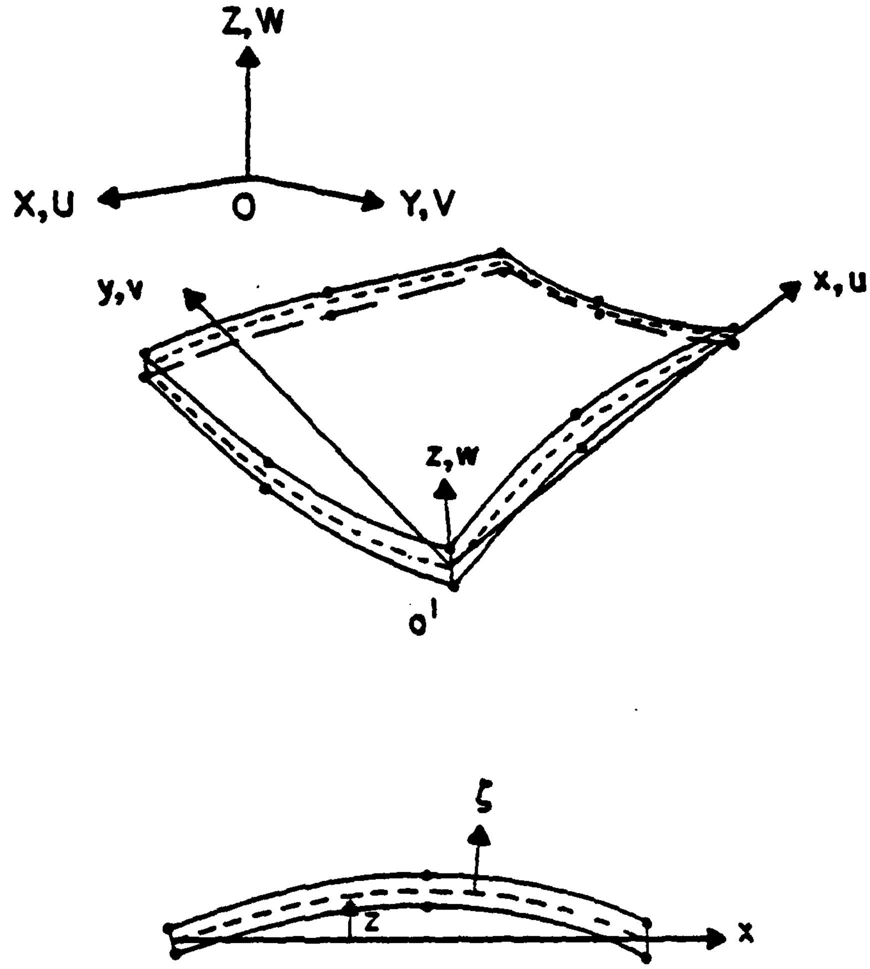



Figure 1. *Undeformed Geometry of a Shell Element.*

The resulting expressions become, after linearization,

$$e_{xx} = u,_x(1 - \zeta Z,_{xx}) - \zeta Z,_{xy}v,_x + Z,_x w,_x$$

$$e_{yy} = v,_y(1 - \zeta Z,_{yy}) - \zeta Z,_{xy}u,_y + Z,_y w,_y$$

$$e_{\zeta\zeta} = w,_\zeta - Z,_x u,_\zeta - Z,_y v,_\zeta$$

$$2e_{y\zeta} = v,_\zeta(1 - \zeta Z,_{yy}) + w,_y$$

$$- Z,_x u,_y + Z,_y(w,_\zeta - v,_y) - \zeta Z,_{xy}u,_\zeta$$

$$2e_{x\zeta} = u,_\zeta\ (1 - \zeta Z,_{xx}) + w,_x$$

$$- Z,_y v,_x + Z,_x(w,_\zeta - u,_x) - \zeta Z,_{xy}v,_\zeta$$

$$2e_{xy} = u,_y(1 - \zeta Z,_{xx}) + v,_x(1 - \zeta Z,_{yy})$$

$$+ Z,_x w,_y + Z,_y w,_x - \zeta Z,_{xy}(u,_x + v,_y). \qquad (8)$$

The behavioral constraints leading to thin plate or shell response are expressed directly in the form

$$e_{x\zeta} = e_{y\zeta} = e_{\zeta\zeta} = 0\ . \qquad (9)$$

Furthermore, the condition of vanishing normal stress through the thickness is imposed exactly, and the remaining stress-strain relation is one of plane stress,

$$\begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \begin{bmatrix} e_{xx} \\ e_{yy} \\ 2e_{xy} \end{bmatrix} . \qquad (10)$$

Equation 10 is written symbolically in the form

$$\underset{\sim}{\sigma} = \underset{\sim}{D}\underset{\sim}{e} \tag{11}$$

in which case the potential energy of a shell is simply

$$\Pi_\rho = \frac{1}{2}\int_V \underset{\sim}{e}^T \underset{\sim}{D}\underset{\sim}{e}\, dV - \int_{\partial V} \underset{\sim}{u}^T \underset{\sim}{t}\, dA. \tag{12}$$

Here $\partial V$ represents the shell boundary, and $\underset{\sim}{t}$ the components of the applied mechanical forces.

To complete the theoretical formulation of the element, the constraints summarized in Equation 9 must be introduced. This is accomplished by introducing the penalty functional,

$$F = \frac{1}{2}\sum_{i=1}^{M}\sum_{j=1}^{3} W_{ij} e_{j\zeta}^2(\vec{r}_{ij}) \ . \tag{13}$$

In Equation 13, $\vec{r}_{ij}$ and $W_{ij}$ represent collocation points and weights, which may be chosen independently for each of the transverse strains. The weights $W_{ij}$ play the role of penalty factors, and as $W_{ij} \rightarrow \infty$ the transverse strain constraints are enforced exactly at each of the points $\vec{r}_{ij}$. In practice, the magnitudes of the $W_{ij}$ are chosen to provide adequate satisfaction of the constraints and good numerical conditioning simultaneously for the practical range of element length-to-thickness ratios[6]. The element basis is simply the augmented functional

$$\Pi_\rho' = \frac{1}{2}\int_V \underset{\sim}{e}^T \underset{\sim}{D}\underset{\sim}{e}\, dV - \int_{\partial V} \underset{\sim}{u}^T \underset{\sim}{t}\, dA + \frac{1}{2}\sum_{i=1}^{M} \underset{\sim}{\gamma}_i^T \underset{\sim}{W} \underset{\sim}{\gamma}_i \tag{14}$$

in which

$$\underset{\sim}{\gamma}_i^T = \lfloor e_{x\zeta}(\vec{r}_{ix}) \ e_{y\zeta}(\vec{r}_{iy}) \ e_{\zeta\zeta}(\vec{r}_{i\zeta}) \rfloor \tag{15}$$

and $\underset{\sim}{W}$ is the matrix of weighting factors $W_{ij}$.

In thermal stress analysis, the constitutive relation (Equation 11) is replaced by

$$\underset{\sim}{\sigma} = \underset{\sim}{D}(\underset{\sim}{e} - \underset{\sim}{\alpha}T) \tag{16}$$

which leads to the energy

$$\Pi_\rho' = \frac{1}{2}\int_V \underset{\sim}{e}^T\underset{\sim}{D}\underset{\sim}{e}\,dV - \int_V \underset{\sim}{e}^T\underset{\sim}{D}\underset{\sim}{\alpha}T\,dV$$

$$- \int_{\partial V} \underset{\sim}{u}^T\underset{\sim}{t}\,dA + \frac{1}{2}\sum_{i=1}^{M} \underset{\sim}{\gamma}_i^T\underset{\sim}{W}\gamma_i \ . \tag{17}$$

The second term of Equation 17 leads to the consistent definition of the thermal "loading" induced by a temperature change $T(x,y,\zeta)$ from the temperature in the reference state.

For dynamic calculations, the kinetic energy is formed directly in three-dimensional form,

$$T = \frac{1}{2}\int_V \rho(\dot{\vec{R}} - \dot{\vec{r}})\cdot(\dot{\vec{R}} - \dot{\vec{r}})\,dV. \tag{18}$$

From Equations 3 and 6,

$$T = \frac{1}{2}\int_V \rho(\dot{u}^2 + \dot{v}^2 + \dot{w}^2)\,dV. \tag{19}$$

Since the potential and kinetic energies formulated above involve derivatives no higher than first order, the finite element approximation must satisfy the continuity of displacements (but not bending slopes) between adjacent elements. In the present study, the simplest permissible form of approximation, based upon linear polynomial shape functions, is used. The resulting element is shown in Figure 2. Despite the simplicity of the approximation, the element exhibits high accuracy in both displacement and stress computations, and is particularly straightforward to use in comparison with many other shell finite elements. Linear and nonlinear analyses which demonstrate the accuracy and economy of the present shell element formulation can be found in Reference 6.

## 2.2 SANDWICH CORE ELEMENTS

A fully three-dimensional formulation is used herein to obtain sandwich core elements compatible with the face sheet elements described in Paragraph 2.1. Define the strain and stress vectors

$$e^T = \lfloor e_{xx}\ e_{yy}\ e_{zz}\ 2e_{yz}\ 2e_{xz}\ 2e_{xy} \rfloor \tag{20}$$

and

$$\underset{\sim}{\sigma}^T = \lfloor \sigma_{xx}\ \sigma_{yy}\ \sigma_{zz}\ \sigma_{yz}\ \sigma_{xz}\ \sigma_{xy} \rfloor . \tag{21}$$

Here the strains are defined by

$$e_{ij} = \frac{1}{2}(u_{i,j} + u_{j,i}). \tag{22}$$

The stress-strain relation is (see Reference 1, Paragraph 2.2)

$$\underset{\sim}{\sigma} = \underset{\sim}{D}(\underset{\sim}{e} - \underset{\sim}{\alpha}T) \tag{23}$$

and the potential energy can be expressed as

$$\Pi_\rho = \frac{1}{2}\int_V \underset{\sim}{e}^T \underset{\sim}{D}\underset{\sim}{e}\, dV - \int \underset{\sim}{e}^T \underset{\sim}{D}\underset{\sim}{\alpha} T\, dV - \int_{\partial V} \underset{\sim}{u}^T \underset{\sim}{t}\, dA. \tag{24}$$

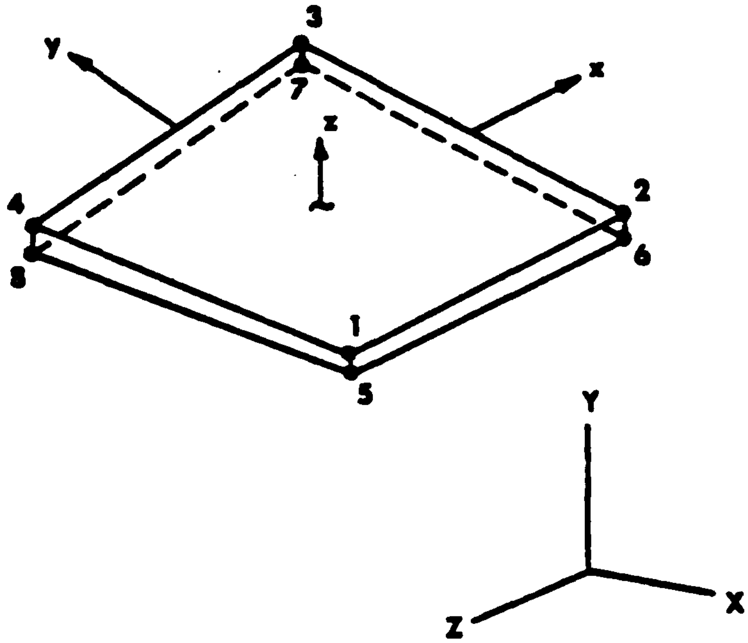



Figure 2. Trilinear Shell Finite Element.

Similarly, the kinetic energy is

$$T = \frac{1}{2} \int_V \rho(\dot{u}^2 + \dot{v}^2 + \dot{w}^2)\, dV. \tag{25}$$

The above energy expressions are discretized using a linear polynomial displacement field in isoparametric coordinates, for compatibility with the face sheet elements. The resulting finite element has eight nodes (Figure 3) and 24 degrees of freedom.

In the approximation of bending response, the above eight-node element is notoriously poor when the potential energy (Equation 24) is evaluated exactly[6]. However, the stabilizing effect of surrounding face sheet and stiffener elements or closeouts permits the element energy (i.e., the stiffness matrix) to be computed using a reduced integration scheme (one point Gaussian quadrature). This single-point integration, though inexact, is sufficient for convergence[7] and permits a much-improved simulation of the panel bending response over the conventional eight-node element.

## 2.3 FACE SHEET AND FULL-DEPTH STIFFENERS

Discrete stiffener elements for use with the present face sheet and core layer elements are obtained using a penalty function approach similar to that employed in Paragraph 2.1, in two dimensions rather than three. The resulting element is displacement-compatible with the remaining elements, represents axial and primary bending effects well, and may be arbitrarily oriented in three-dimensional space.

Both types of stiffeners are shown in Figure 4. In each case, the stiffener is considered to be relatively small in width, so that all stresses are approximately uniform in the width direction. This assumption causes inplane bending effects to be ignored, which is consistent with the resolution provided by the face sheet discretization. Normal stresses in the width direction

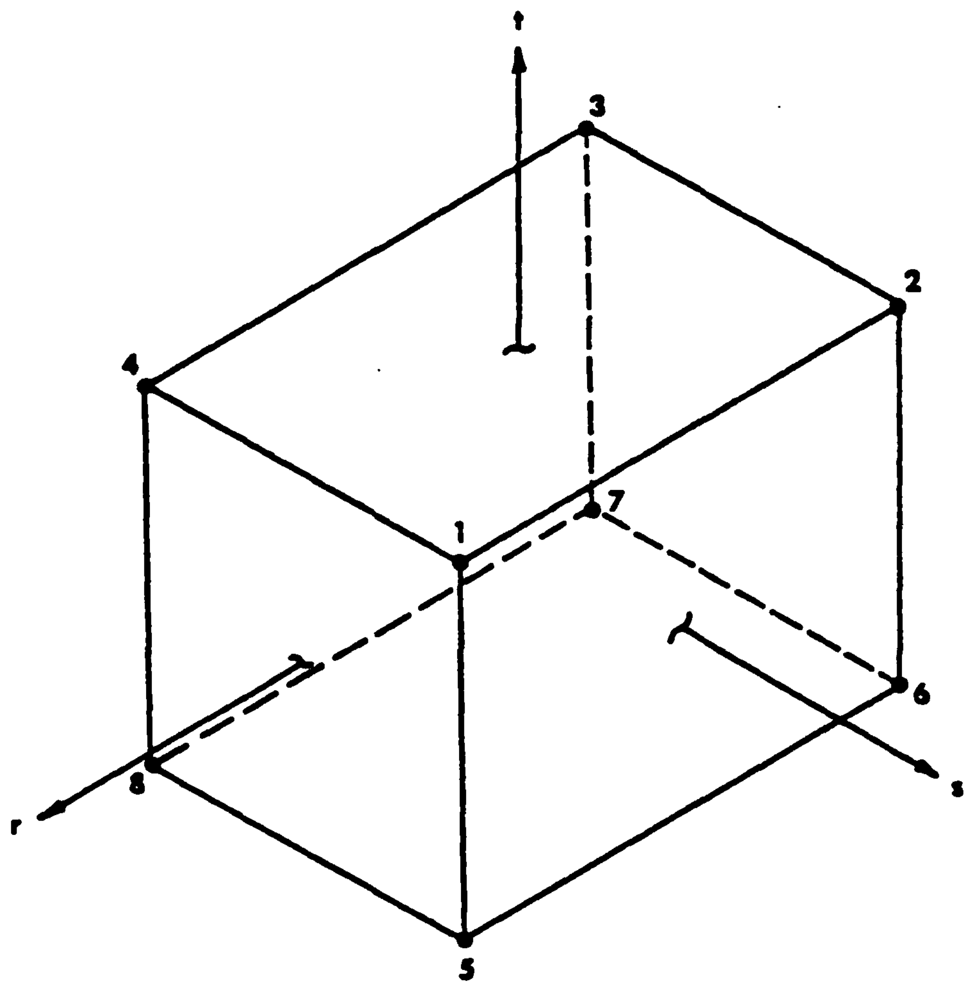



Figure 3. Sandwich Core Finite Element.

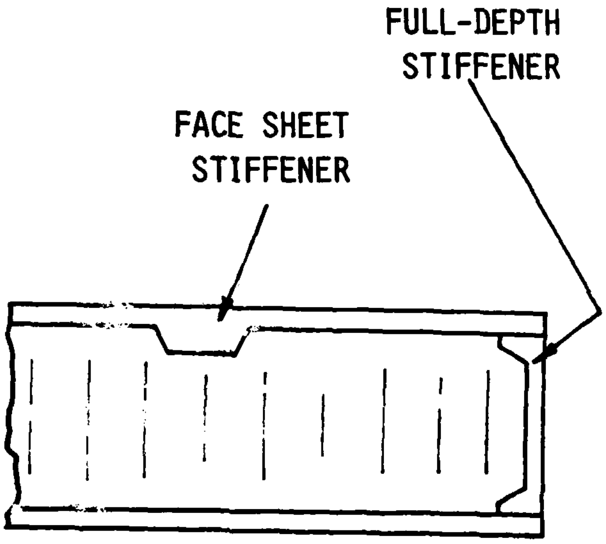



Figure 4. Face Sheet and Full-Depth Stiffeners.

are also neglected; thus, the stiffener is subject to a plane state of stress in the remaining dimensions, although arbitrary warping and twisting of the element is permitted.

The strain energy of a typical element involves only the longitudinal strains, which give rise to both axial and bending forces in the deformed state. Remaining components of strain (transverse shear and normal strain in the depth direction) are suppressed through the use of penalty function constraints (see Paragraph 2.1). Denoting the longitudinal direction by x, the potential energy is simply[6]

$$\Pi_\rho = \frac{1}{2}\int_V E e_{xx}^2 dV - \int_V E\alpha^T e_{xx} dV - \int_{\partial V} \underset{\sim}{u}^T \underset{\sim}{t} dA \tag{26}$$

where E is the elastic modulus, and the kinetic energy is

$$T = \frac{1}{2}\int_V \rho(\dot{u}^2 + \dot{v}^2 + \dot{w}^2) dV. \tag{27}$$

To enforce the remaining constraints

$$e_{zz} = e_{xz} = 0, \tag{28}$$

introduce the penalty functional

$$F = \frac{1}{2} W_x e_{xz}^2(\vec{r}_x) + \frac{1}{2} W_z e_{zz}^2(\vec{r}_z) \tag{29}$$

in which $W_x$, $W_z$ are the constraint weights and $\vec{r}_x$, $\vec{r}_z$ are the corresponding collocation points. Analogous to the procedure of Paragraph 2.1, the functional F is added to the potential energy (Equation 26) to form the constrained energy on which the element stiffenss matrix is based:

$$\Pi_\rho' = \frac{1}{2}\int_V Ee_{xx}^2 dV - \int_V E\alpha Te_{xx} dV$$

$$+ \frac{1}{2} W_x e_{xz}^2 (\vec{r}_x) + \frac{1}{2} W_z e_{zz}^2 (\vec{r}_z)$$

$$- \int_{\partial V} \underset{\sim}{u}^T \underset{\sim}{t} dV. \qquad (30)$$

The above potential and kinetic energies are last in discrete form through a linear approximation of the displacements $\underset{\sim}{u}$ within individual elements. The resulting finite elements, shown in Figure 5, have four nodes and twelve degrees of freedom. For this order of discretization, the element strain energy may be evaluated exactly using a four-point (2 by 2 Gauss) integration rule. The transverse strain constraints (Equation 28) are each applied at a single point corresponding to the element centroid location.

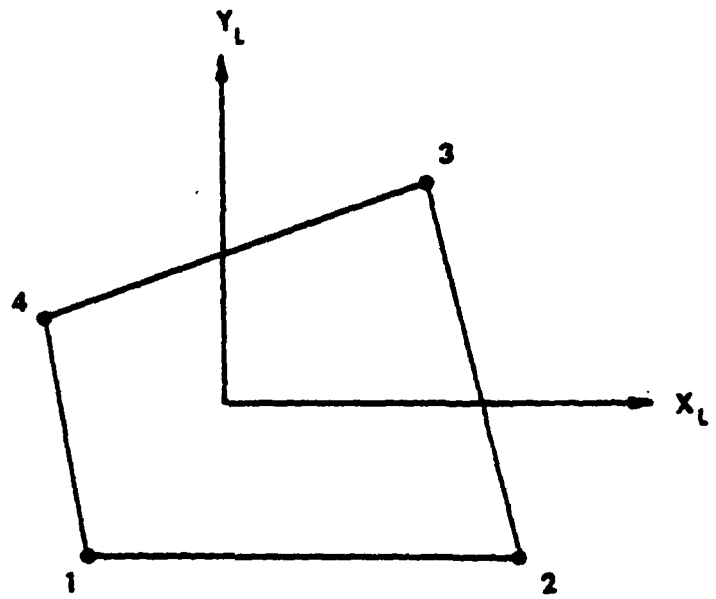



Figure 5. Stiffener Finite Element in Local Coordinates.

## SECTION 3
## NUMERICAL SOLUTION TECHNIQUES

The methods used in solving the governing matrix equations for a complete finite element model are described in this Section. Two general types of solutions are presented: linear simultaneous equations, for static analysis with mechanical and/or thermal loading; and generalized eigenvalue systems, for use in natural vibration analysis.

### 3.1 LINEAR STATIC ANALYSIS

In static analysis, minimization of the discrete potential energy for an assemblage of elements leads to the matrix equation

$$\underset{\sim}{K}\underset{\sim}{U} = \underset{\sim}{T} \tag{31}$$

in which $\underset{\sim}{K}$ is the stiffness matrix, $\underset{\sim}{U}$ are the (unknown) nodal displacements, and $\underset{\sim}{T}$ is the load vector including both mechanical and thermal generalized forces. A direct solution is obtained by a Gauss-Doolittle factorization of $\underset{\sim}{K}$, followed by forward and backward substitution. The decomposition of $\underset{\sim}{K}$ is of the form

$$\underset{\sim}{K} = \underset{\sim}{L}\underset{\sim}{D}\underset{\sim}{L}^T. \tag{32}$$

Here $\underset{\sim}{L}$ is a unit lower triangular matrix,

$$\ell_{ij} = \begin{cases} \text{(nonzero)}, & i>j \\ 1, & i=j \\ 0, & i<j \end{cases} \tag{33}$$

and $\underset{\sim}{D}$ is a strictly diagonal matrix. Next let

$$\underset{\sim}{Z} = \underset{\sim}{D}\underset{\sim}{L}^T\underset{\sim}{U} \tag{34}$$

and the relation

$$\underset{\sim}{L}\underset{\sim}{Z} = \underset{\sim}{T} \tag{35}$$

can be solved for $\underset{\sim}{Z}$ by a forward substitution. From Equation 34,

$$\underset{\sim}{L}^T\underset{\sim}{U} = \underset{\sim}{D}^{-1}\underset{\sim}{Z} \tag{36}$$

which can then be solved for $\underset{\sim}{U}$ in a backward substitution step.

Having solved for the nodal displacements $\underset{\sim}{U}$ over the entire finite element model, a complete vector of displacements can be extracted for each individual element. The element displacement vector, along with the shape functions for an element, completely describe the state of displacement (and, therefore, strain and stress) within each element of the model.

## 3.2 NATURAL FREQUENCY SOLUTION

In natural vibration analysis, the discretization of both the potential and kinetic energies and the assumption of harmonic motions yield the governing system of equations

$$\underset{\sim}{K}\underset{\sim}{U} = \omega^2\underset{\sim}{M}\underset{\sim}{U} \quad . \tag{37}$$

Here $\underset{\sim}{K}$ and $\underset{\sim}{M}$ are the stiffness and mass matrices, respectively, $\underset{\sim}{U}$ is the nodal displacement vector, and $\omega$ is the circular frequency of vibration.

It should be noted in Equation 37 that the mode shape $\underset{\sim}{U}$ describes only the relative displacements of the mesh points ($a\underset{\sim}{U}$, where a is a constant, is a solution as well). If matrices $\underset{\sim}{K}$ and $\underset{\sim}{M}$ are of order N, then there exist N-solutions of Equation 37: $\omega_i$, $\underset{\sim}{U}_i$; i=1,2,.....,N. Usually only the lowest several frequencies and mode shapes are of interest, and thus a vector iteration procedure is most appropriate for solving the system. An approach based upon simultaneous vector iteration[8] is used herein.

Factoring the stiffness matrix by Choleski's method gives

$$\underset{\sim}{K} = \underset{\sim}{L}\underset{\sim}{L}^T. \tag{38}$$

Equation 37 can now be restated as the standard eigenvalue problem

$$\underset{\sim}{A}\underset{\sim}{V} = \lambda \underset{\sim}{V} \tag{39}$$

in which

$$\underset{\sim}{A} = \underset{\sim}{L}^{-1}\underset{\sim}{M}\underset{\sim}{L}^{-T} \tag{40}$$

$$\underset{\sim}{V} = \underset{\sim}{L}^T\underset{\sim}{U} \tag{41}$$

and

$$\lambda = \frac{1}{\omega^2} . \tag{42}$$

The eigenvalue solution is performed using a set of m trial vectors, where m is greater than the number of frequencies desired but is generally much less than N. These trial vectors, which are arranged in the partial modal matrix

$$\underset{\sim}{\Phi} = [\underset{\sim}{V}_1 \ \underset{\sim}{V}_2 \ . \ . \ . \ . \ \underset{\sim}{V}_m] \tag{43}$$

gradually converge to the first in eigenvectors (mode shapes) as the iteration proceeds.

The central step in the simultaneous iteration is the function of the m by m interaction matrix

$$\underset{\sim}{B} = \underset{\sim}{\Phi}^T\underset{\sim}{A}\underset{\sim}{\Phi}. \tag{44}$$

When the iteration has converged, $\underset{\sim}{B}$ is a diagonal matrix since all of the vectors $\underset{\sim}{V}_i$ are orthogonal with respect to A. Before the solution has converged, an approximate eigenvalue analysis of $\underset{\sim}{B}$ is performed at each iteration to generate improved estimates of the eigenvlaues and to modify the iteration matrix $\underset{\sim}{\Phi}$. Details of the implementation of this procedure may be found in Reference 8.

## SECTION 4
## COMPUTER PROGRAM DESCRIPTION

The finite element discretization and numerical solution procedures summarized in Sections 2 and 3 are implemented in a computer program which permits the interactive solution of static and natural frequency problems for sandwich composites. Auxiliary programs are used in conjunction with the primary analysis program to perform the tasks of data input and generation, and of presentation of results.

### 4.1 PROGRAM FUNCTIONS AND ORGANIZATION

The system of computer programs which are used for the interactive analysis of sandwich panels are pictured in Figure 6. A data generation program is used to prepare data interactively for use in the main analysis program. The sandwich analysis input data (file INPDAT) conforms to the input data formats of Reference 1, which in turn are patterned after those of the MAGNA general purpose finite element program[4]. Thus, the geometry plotting capabilities described in Reference 9 may be used to obtain line drawings of the generated mesh for documentation purposes when required. Once the requested analysis has been performed, a summary of results is stored on disc (file POSTFL) for deformed geometry plotting[9], listing by separate output formatting program, or listing on the line printer.

### 4.2 INTERACTIVE OPERATION

The interactive programs described here are of three types: data input and generation; static or natural frequency analysis; and tabular output. Operating instructions for compatible pre- and post-processing graphics programs are given in Reference 9.

Data generation is performed using a minimal amount of input from the user concerning the panel geometry, properties, supports, loading, temperature distribution and mesh divisions. In particular, the following data are requested:

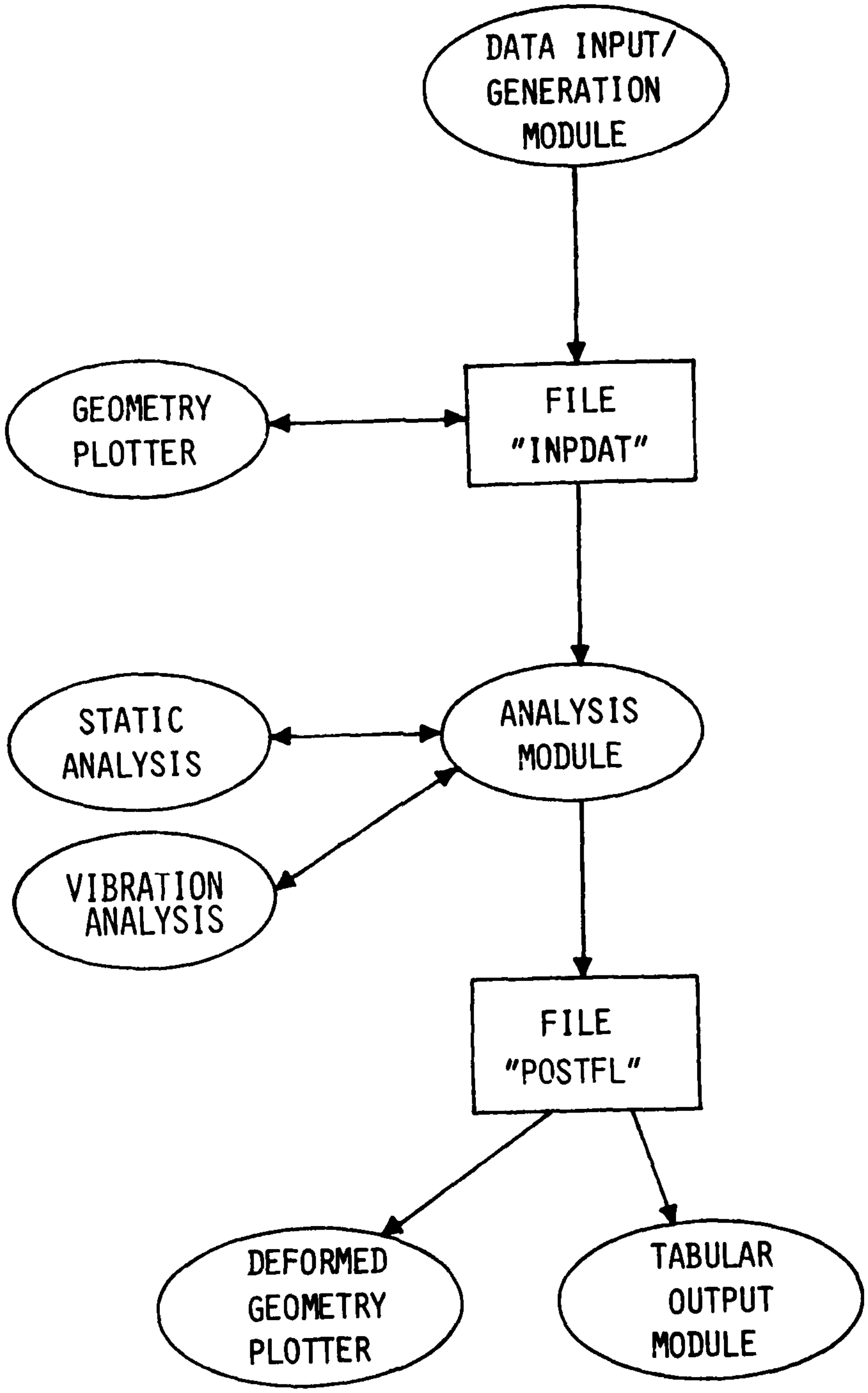



Figure 6. Computer Program Organization.

1. ANALYSIS TYPE: static or natural frequency solution.

2. NUMBER OF MODES: number of natural frequencies and mode shapes to be solved in vibration analysis.

3. TYPE OF LOADING: mechanical or thermal loads (or both) in static analysis.

4. PANEL DIMENSIONS A,B: planform dimensions of the sandwich panel in the x,y directions. At present, the data generation program is restricted to flat, rectangular panels.

5. LAYER THICKNESS TL,TC,TU: thicknesses of lower face sheet, core layer, and upper face sheet.

6. FACE SHEET PROPERTIES E,NU,ALFA,RHO: Young's modulus, Poisson's ratio, coefficient of thermal expansion and density (requested for each face sheet in turn).

7. CORE PROPERTIES GX,GY,ALFAX,ALFAY,RHO: shear moduli, coefficients of thermal expansion and density for the core layer.

8. NUMBER OF STIFFENERS NSX,NSY: number of stiffening members in the x and y directions, respectively. Both face sheet and full depth stiffeners are included.

9. STIFFENER LOCATIONS X(OR Y), ITYPE: for each stiffening member, the constant value of x or y at which the member is located is requested; valid x-values are between O and A, and valid y-values are between O and B. For each member, the type is also requested (lower face, upper face, full depth).

10. STIFFENER PROPERTIES E, ALFA, T, RHO: for each stiffener, these are the values of the Young's modulus, the coefficient of thermal expansion, the stiffener width, and the mass density.

11. SUPPORT CONDITIONS ISO,ISA,JSO,JSB: for each of the edges of the panel (x = o,a; y = o,b), a boundary condition must be specified. Support conditions which can be chosen independently at each panel edge include: free, simple support, clamped, symmetric and antisymmetric.

12. PRESSURE LOADINGS PL,PU: in linear static analysis, pressures may be specified on the lower or upper face sheet surfaces, or both. Note that all pressure values are positive upward.

13. POINT LOADINGS XP,YP,FORCE: location and magnitude of point loadings for static analysis.

14. NUMBER OF ELEMENTS PER EDGE NEX,NEY: this data defines the degree of mesh refinement in the model by dividing the sandwich panel into a rectangular grid of NEX by NEY finite elements. Usually a moderate number of elements (4-6 per edge) provides a good balance of accuracy and solution cost.

15. GRID POINT LOCATIONS XG (OR YG): along each side of the panel, the locations of grid lines may be specified to generate nonuniform (graded) meshes. NEX+1 locations must be specified along the x-axis, and NEY+1 locations on the y-axis. Note that each of the stiffener locations specified in Item 9 must correspond to a grid line location.

In addition to geometric parameters and other data defined in the interactive mode, temperature data must be supplied if thermal stress analysis is to be performed. By default, the data generator will create temperature data for a uniformly heated panel (T = 100°, in consistent units). To specify a different temperature distribution, the user must supply a separate subroutine to generate the required data $T = T(x,y,z)$. This user-written subroutine has the form shown in the following example, for which the temperature distribution is

$$T(x,y,z) = 1.2Ze^{-0.028x} \quad . \tag{45}$$

The appropriate user subroutine (which always has the name UTEMPS) is

```
SUBROUTINE UTEMPS (X,Y,Z,T)
T = 1.2*Z*EXP(-0.028*X)
RETURN
END .
```

User subroutine UTEMPS is called once for every generated point in the finite element model.

The analysis is next performed by simply loading and executing the analysis module. While the final results of an analysis are written to the file POSTFL for later processing, certain information, including diagnostic and warning messages, is output directly at the user's terminal. In the event that a problem becomes too large for the interactive program, the analysis can be performed in the batch mode using the program of Reference 1, since the input data are fully compatible.

Following the analysis step, tabular results may be printed by simply executing the output module. Analysis results provided in table form include deflection data on selected grid lines, and strain and stress data at individual element centroids. Alternatively, the file POSTFL may be used as input to graphics programs[9] which produce deformed geometry plots for each visualization of the problem results.

## SECTION 5
## SAMPLE ANALYSES

Typical analyses performed using the solution techniques described in Sections 2 and 3 are presented in this section. Static problems involving mechanical and thermal loading, as well as natural frequency solutions, are discussed. The operation of the analysis program is demonstrated in greater detail in the sample terminal session included in the Appendix.

### 5.1 STATIC ANALYSIS WITH MECHANICAL AND THERMAL LOADS

To demonstrate the static analysis option, a rectangular panel previously studied by Monforton[10] and Kan and Huang[11] is considered. The flat sandwich plate has the following geometry and properties:

| | |
|---|---|
| Planform: | $a=b = 50$ inches |
| Thicknesses: | $t_f = 0.015$ inches |
| | $t_c = 1.000$ inches |
| Face Material: | $E_f = 10.5 \times 10^6$ lb/in$^2$ |
| | $\nu_f = 0.30$ |
| | $\alpha_f = 12.4 \times 10^{-6}$ in/in°F |
| Core Material: | $G_c = 50000.$ lb/in$^2$ |
| | $\alpha_c = 12.4 \times 10^{-6}$ in/in°F . |

Using these parameters, a static solution is performed for the case of a fully clamped boundary and uniform lateral pressure (Figure 7). Due to symmetry of the problem only one quarter of the panel is modeled. The mesh consists of a 5 by 5 grid of elements in each layer, for a total of 260 unconstrained degrees of freedom. For a unit lateral pressure, the computed central displacement is $\omega_c = 0.113$ inches. This value is in good agreement with previously published results; for example, linearization of the results of Reference 11 gives $\omega_c = 0.09497$ inches.

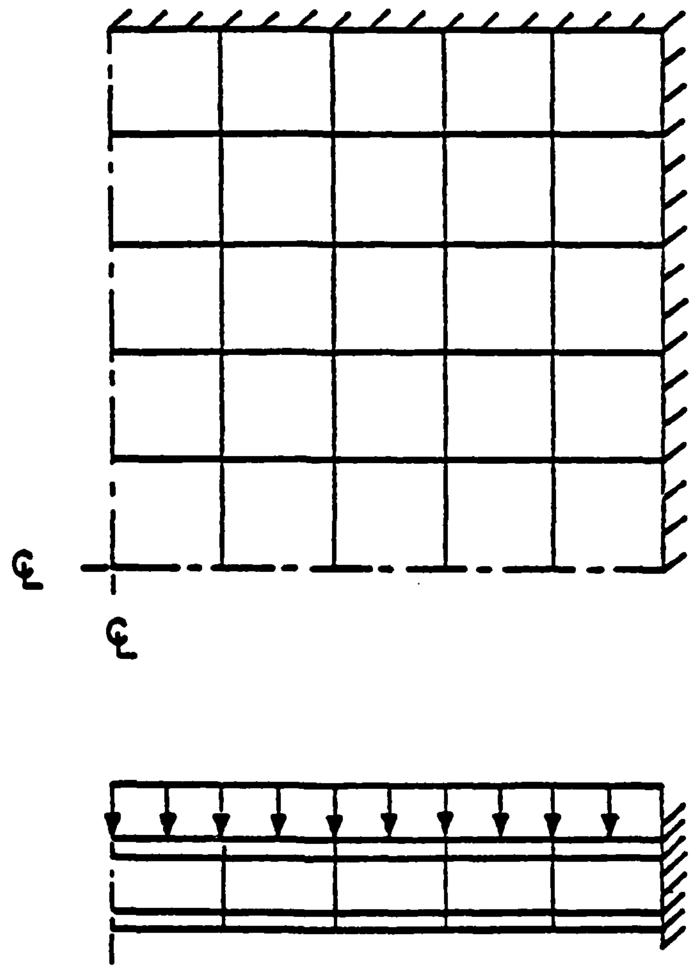



Figure 7. Clamped Panel Under Uniform Pressure.

A static analysis has also been performed for the above panel considering thermal loading only. In this case, the panel is assumed to be simply supported. The temperature distribution is such that the two face sheets are at opposite uniform temperatures, and the core experiences a linear thermal gradient. The following user subroutine is used to define the thermal field:

```
SUBROUTINE UTEMPS (X,Y,Z,T)
T = 200.*Z
IF (ABS(Z).GT.0.5)T = 100.*T/ABS(T)
RETURN
END .
```

This antisymmetric temperature field produces a pure moment type of loading, and thus a bending response. The resulting deformed shape of the panel is shown in Figure 8. Typical Stress results (in this case the core shear stress $T_{xz}$) are depicted in graphical form in Figure 9.

## 5.2 NATURAL FREQUENCY ANALYSIS

A simply supported sandwich panel is considered to determine its free vibration response. The following geometric and material parameters are used:

| | |
|---|---|
| Planform: | $a = 62.25$ inches |
| | $b = 43.50$ inches |
| Thicknesses: | $t_f = 0.072$ inches |
| | $t_c = 1.856$ inches |
| Face Sheets: | $E_f = 10. \times 10^6$ lb/in$^2$ |
| | $\nu_f = 0.33$ |
| | $\rho_f = 0.000244$ lb-sec$^2$/in$^4$ |
| Core: | $G_c = 30000.$ lb/in$^2$ |
| | $\rho_c = 0.0000123$ lb-sec$^2$/in$^4$ . |

The finite element model of one quarter of the panel is shown in Figure 10. Thirty-six elements are used in each layer of the model.

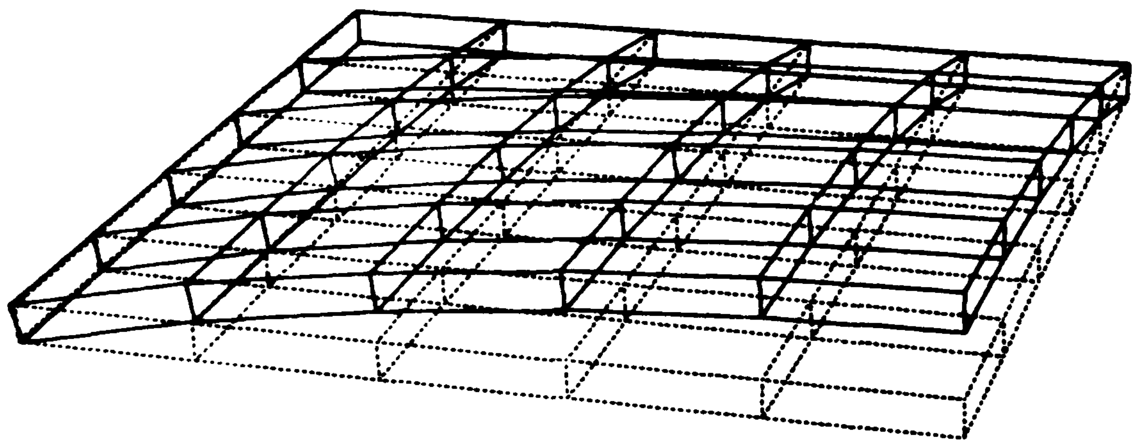


Figure 8. Deformed Geometry of Square Panel Under Antisymmetric Thermal Loading.

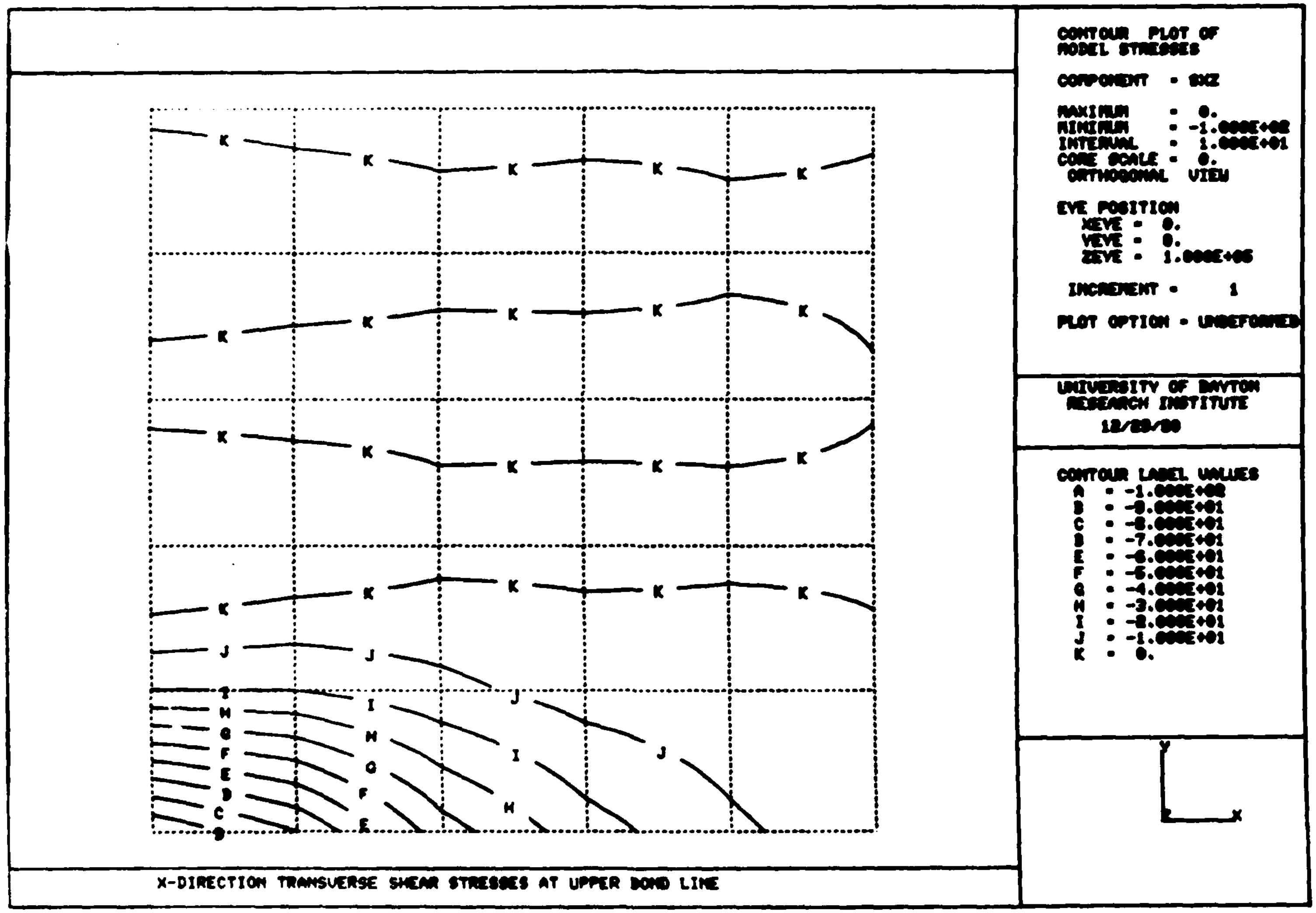



Figure 9. Core Shear Stresses for Sandwich Panel with Antisymmetric Temperature Distribution.

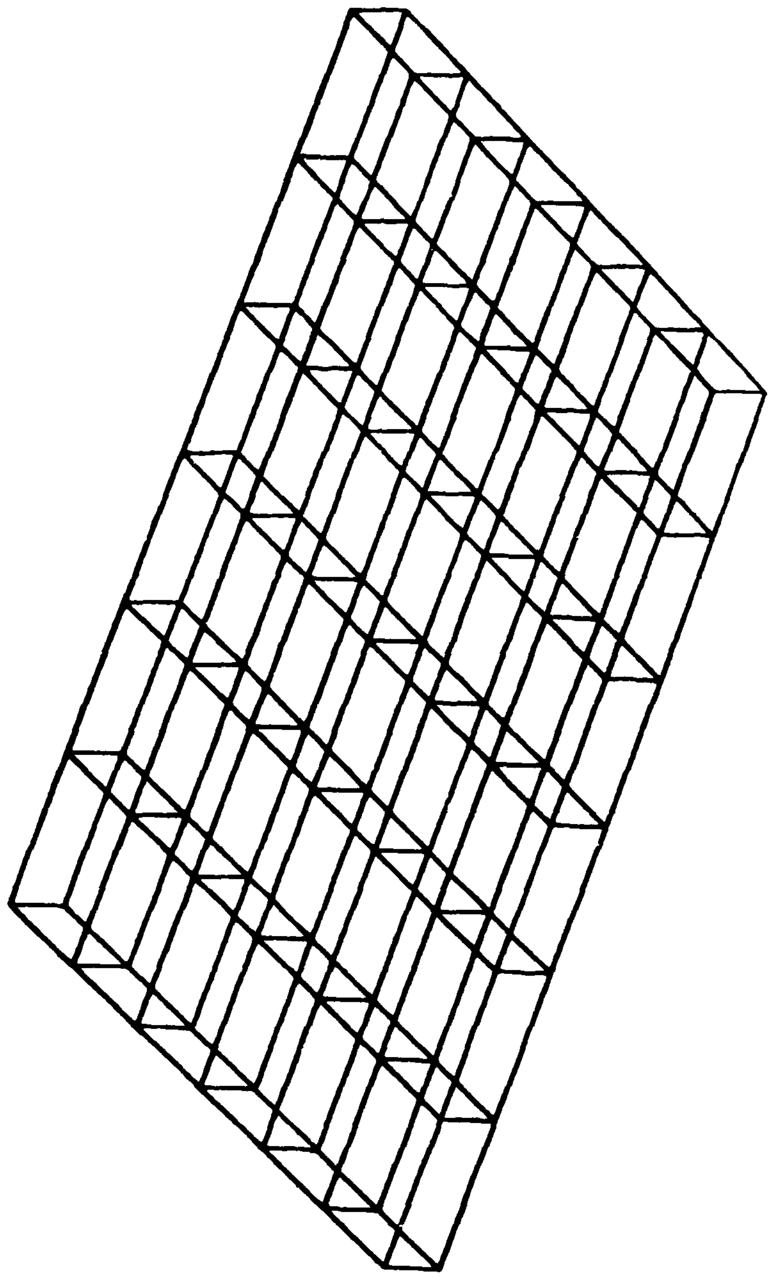


Figure 10. Sandwich Panel for Natural Frequency Analysis.

The solution for the fundamental frequency of the panel using the model shown yields $\omega_1 = 174$ Hz. This value compares well with the classical analytical solution[12] of $\omega_1 = 170$ Hz.

A second solution has also been performed for a similar panel of larger aspect ratio. Panel data for this example is identical to the problem above, with the following exceptions:

$$b = 34.00 \text{ inches}$$
$$t_f = 0.063 \text{ inches}$$
$$t_c = 1.874 \text{ inches.}$$

In this case, the computed frequency of 240 Hz is also in good agreement with a closed-form solution[12] of $\omega_1 = 250$ Hz.

## SECTION 6
## SUMMARY AND CONCLUSIONS

A finite element approach for the static stress analysis of sandwich panels under mechanical and thermal loadings, and for the natural frequency analysis of sandwich, has been described. Layers of a sandwich panel are modeled explicitly, so that unequal face sheets, stiffening members, and other design details are represented with relatively high accuracy. Due to the finite element basis of the present methodology, rather arbitrary geometries and variations in mechanical properties are easily considered.

The methods described are suitable for interactive use, to facilitate parametric and design studies. Data generation utilities, which permit the definition of simple panel geometries from a minimal amount of input, are provided to support this mode of usage. Generated input is compatible with that of a comparison program[1] with larger capability. Results are output in a form suitable for plotting using existing computer graphics programs.[9]

The developments described herein can be improved to provide more detailed and comprehensive analytical capabilities (e.g., curved panels, arbitrary stiffener shapes, linear buckling analysis, etc.). However, in their present form, the methods presented represent a valuable capability for practical design analysis of sandwich composite structures.

## APPENDIX

## SAMPLE INTERACTIVE TERMINAL SESSION

A typical interactive analysis session using the sandwich analysis program described herein is reproduced in this Appendix. This sample session demonstrates the procedures for accessing and executing the data generation, analysis and plotting programs, and illustrates the types of output which can be obtained from the analysis.

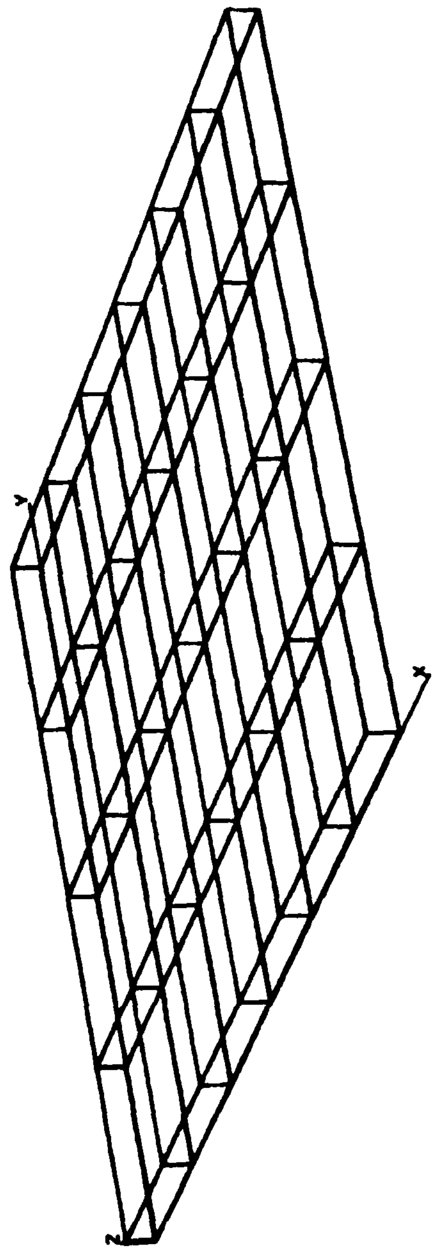
The analysis procedure is initiated by accessing a stored control language procedure file (denoted by "PROCFIL" in the example), and issuing a command to start the procedure:

```
BEGIN,SWICH,PROFIL.
```

This command automatically invokes the data generation, analysis and tabular output modules. By appending the keyword "PLOT" to the BEGIN command, a geometry plotting utility is also executed before and after the analysis module. The "PLOT" option is used in the sample session to clarify the problem geometry and results.

```
COMMAND- ATTACH,PROCFIL,PROCFIL,ID=DB10114,MR=1.
 AT CY= 001 SN=AFFDL
COMMAND- BEGIN,SWICH,PROCFIL,PLOT.

 BEGIN DATA GENERATION PHASE

 ANALYSIS TYPE CODES
    1 = STATIC SOLUTION
    2 = NATL. FREQUENCY

 ENTER ANALYSIS TYPE...... 1

 LOADING TYPE CODES
    1 = MECHANICAL LOADS
    2 = THERMAL LOADS
    3 = BOTH

 ENTER LOADING TYPE....... 1

 ENTER PANEL DIMENSIONS
    (A,B)................. 30, 24

 ENTER FACE SHEET AND CORE
 THICKNESSES (TL,TC,TU)... 0.050, 0.875, 0.035

 PROPERTIES FOR FACE SHEET 1
 ENTER E, PR, ALFA, RHO... 10.3E6, 0.3, 0, 0

 PROPERTIES FOR FACE SHEET 2
 ENTER E, PR, ALFA, RHO... 10.3E6, 0.3, 0, 0

 CORE LAYER PROPERTIES
 ENTER GX, GY, ALFAX,
       ALFAY, RHO......... 30000, 50000, 0, 0, 0

 STIFFENING MEMBER DATA

 ENTER THE NUMBER OF
 STIFFENERS WITH X=CONST
 AND Y=CONST (NSX,NSY).... 2, 2

 ENTER THE X-STATIONS AT
 WHICH STIFFENERS ARE
 TO BE LOCATED............ 0, 20

 ENTER THE TYPE OF EACH STIFFENER
    1 = LOWER FACE
    2 = UPPER FACE
    3 = FULL DEPTH........... 3, 2

 ENTER THE Y-STATIONS AT
 WHICH STIFFENERS ARE
 TO BE LOCATED............ 0, 24

 ENTER THE TYPE OF EACH STIFFENER
    1 = LOWER FACE
    2 = UPPER FACE
    3 = FULL DEPTH........... 3, 3

 ARE ALL STIFFENERS TO
 BE ASSIGNED IDENTICAL
 PROPERTIES (Y/N).........Y

 ENTER PROPERTIES
    E, T, RHO, ALFA....... 10.0E6, .1, 0, 0

 BOUNDARY CONDITION CODES
  0 = FREE EDGE
  1 = SIMPLE SUPPORT
  2 = CLAMPED EDGE
  3 = SYMMETRIC EDGE

 ENTER BOUNDARY CODES FOR
 X=0, X=A, Y=0, Y=B....... 2, 3, 0, 0

 ENTER LOWER AND UPPER
 SURFACE PRESSURES PL,PU.. 1, 0

 ENTER THE NUMBER
 OF POINT LOADS..........0

 ENTER THE NUMBER OF
 ELEMENTS IN EACH DI-
 RECTION (NEX,NEY)........6, 4

 ENTER X-AXIS NODE
 LOCATIONS (NEX+1 VALUES):  0, 5, 10, 15, 20, 25, 30

 ENTER Y-AXIS NODE
 LOCATIONS (NEY+1 VALUES): 0, 6, 12, 18, 24
   ENTER THE CHARACTERS PER SECOND .............: 120
     STOP
     023300 MAXIMUM EXECUTION FL.
        .228 CP SECONDS EXECUTION TIME.
   TEKTRONIX OR HP PLOTTER(T,H) ................: T
   MODEL INPUT FILE TYPES:
   1 - INPUT DATA FILE
   2 - POSTPROCESSOR (MPOST) FILE
   ENTER FILE TYPE(1,2) ........................: 1
```

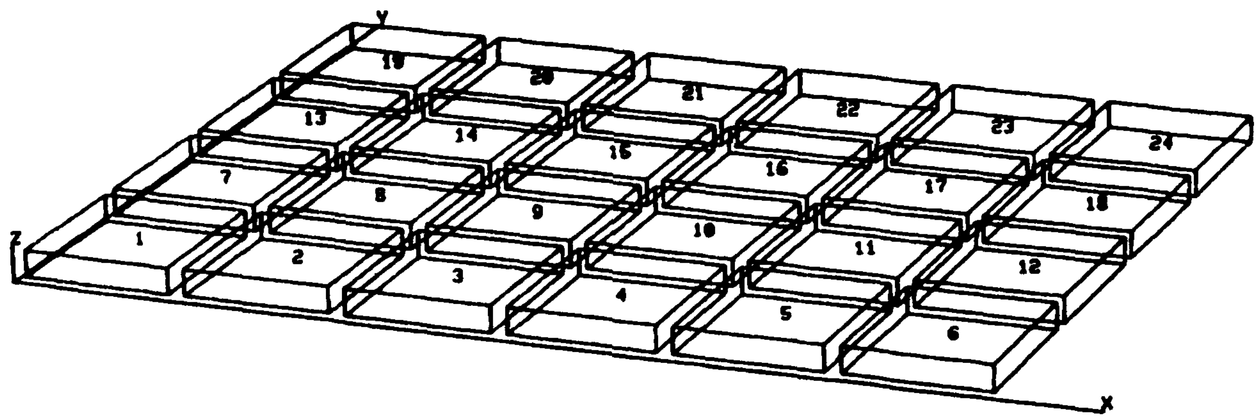
```
  ENTER THE STRUCTURE NUMBER ...................: 1
$HELP
  COMMAND         DESCRIPTION
  $$$$$$$         $$$$$$$$$$$
   AXES            AXES DRAW AND LABEL
   CLIP            CLIP PLANE POSITION
   CUBE            SET MINIMA AND MAXIMA
   DEFAULT         SET DEFAULT VALUES
   DEFORM          DEFORMED MODEL REPRESENTATION
   DRAW            DRAW MODEL
   ELEMENTS        PLOTS ALL OR SELECTED ELEMENTS
   EYE             EYE POSITION
   HELP            LIST ALL COMMANDS
   LABELS          LABELS ELEMENTS AND/OR NODES
   NEW             NEW STRUCTURE
   PROJECTION      PROJECTION TYPE
   REFLECT         REFLECT A PLANE
   ROTATE          ROTATE MODEL ABOUT AXES
   SCALE           SCALE PLOT
   SHRINK          SHRINK ELEMENTS
   STOP            END PROGRAM
   SUMMARY         LIST ALL PARAMETER VALUES
   TIME            PRINT CPU TIME SINCE START OF SESSION
   TRANSLATE       TRANSLATE MODEL FROM ORIGIN
   VERTICAL        VERTICAL AXIS
   ZOOM            ZOOM ON THE MODEL
   SITE            SITE POSITION
$AXES
  PLOT AND LABEL THE AXES?(Y,N) ................: Y
$EYE
  ENTER THE EYE POSITION.
  XEYE, YEYE, ZEYE ............................: 200, -100, 60
$DRAW
```



y
x
z

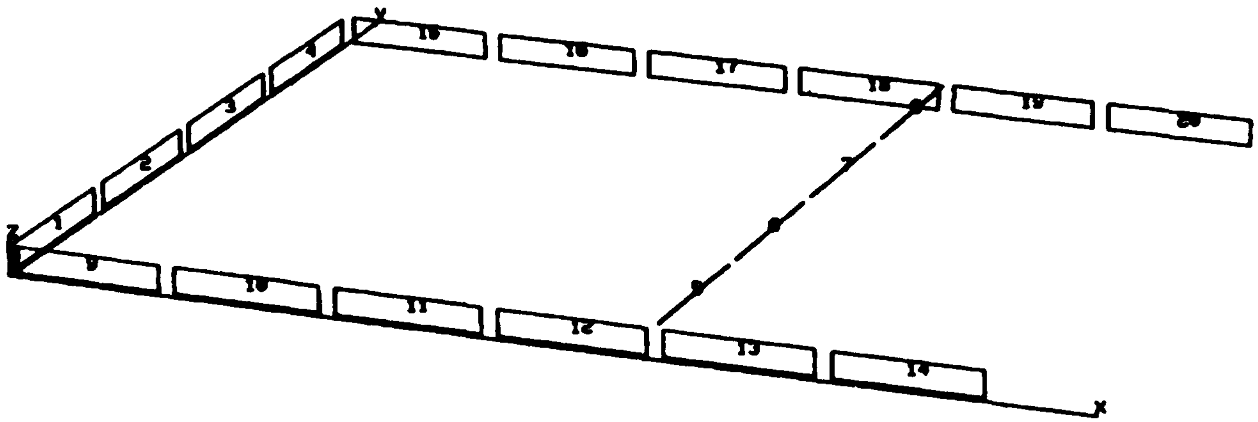
```
$EYE
  ENTER THE EYE POSITION,
  XEYE, YEYE, ZEYE ...............................: 100, -300, 75
$ELEM
  PLOT ALL THE ELEMENTS?(Y,N) ....................: N
  ENTER THE NUMBER OF ELEMENT TYPES TO PLOT ...: 1
  ELEMENT INPUT OPTIONS
  OPTION 1:   RANDOM ELEMENTS(4,13,64,....)
              WHERE FIRST DIGIT = NO. ELEMENTS
  OPTION 2:   RANGE OF ELEMENTS(1,9,2 FOR 1-9,INC OF 2)
  OPTION 3:   PLOT ALL ELEMENTS
  ENTER THE ELEMENT TYPE ........................: 8
  ENTER THE INPUT OPTION(1,2,3) .................: 3
$SHRINK
  ENTER THE SHRINK SCALE(0.-1.) .................: .1
$LABE
  LABEL THE ELEMENTS?(Y,N) ......................: Y
  LABEL THE NODES?(Y,N) .........................: N
$DRAW
```

CORE ELEMENTS ONLY FOR 30 X 24 SYMMETRIC PLATE

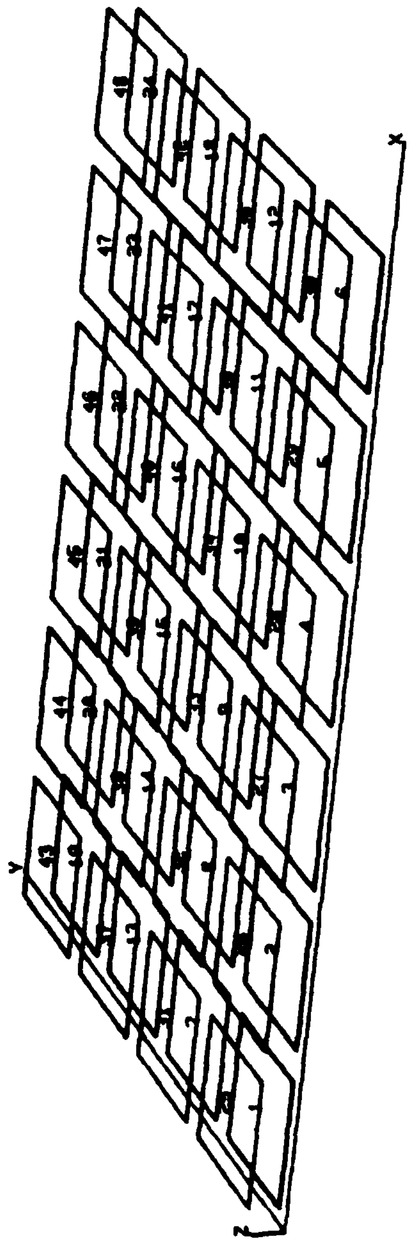


```
$ELEM
  PLOT ALL THE ELEMENTS?(Y,N) ......................: N
  ENTER THE NUMBER OF ELEMENT TYPES TO PLOT ...: 1
  ELEMENT INPUT OPTIONS
  OPTION 1:    RANDOM ELEMENTS(4,13,64,....)
               WHERE FIRST DIGIT = NO. ELEMENTS
  OPTION 2:    RANGE OF ELEMENTS(1,9,2 FOR 1-9,INC OF 2)
  OPTION 3:    PLOT ALL ELEMENTS
  ENTER THE ELEMENT TYPE .........................: 3
  ENTER THE INPUT OPTION(1,2,3) ..................: 3
$DRAW
```

STIFFENER ELEMENTS ONLY FOR 30 X 24 SYMMETRIC PLATE



```
$ELEM
  PLOT ALL THE ELEMENTS?(Y,N) ..................: N
  ENTER THE NUMBER OF ELEMENT TYPES TO PLOT ...: 1
  ELEMENT INPUT OPTIONS
  OPTION 1:   RANDOM ELEMENTS(4,13,64,...)
              WHERE FIRST DIGIT = NO. ELEMENTS
  OPTION 2:   RANGE OF ELEMENTS(1,9,2 FOR 1-9,INC OF 2)
  OPTION 3:   PLOT ALL ELEMENTS
  ENTER THE ELEMENT TYPE ......................: 5
  ENTER THE INPUT OPTION(1,2,3) ...............: 3
$SHRINK
  ENTER THE SHRINK SCALE(0.-1.) ...............: .2
$DRAW
```



```
$STOP

    STOP
    047000 MAXIMUM EXECUTION FL.
     8.573 CP SECONDS EXECUTION TIME.

****************************************

 BEGIN POSTPROCESSING PHASE     *********

****************************************


OUTPUT OPTIONS FOR ANALYSIS DATA  -

 0 - STOP
 1 - LIST LOWER SURFACE DISPL.
 2 - LIST UPPER SURFACE DISPL.
 3 - LIST UPPER FACE STRESSES
 4 - LIST LOWER FACE STRESSES
 5 - LIST UPPER FACE STRAINS
 6 - LIST LOWER FACE STRAINS
 7 - LIST CORE LAYER STRESSES
 8 - LIST CORE LAYER STRAINS
 9 - LIST TIME/FREQUENCY VALS.

ENTER OUTPUT OPTION NUMBER....
    STOP
    051500 MAXIMUM EXECUTION FL.
    13.605 CP SECONDS EXECUTION TIME.
EOI ENCOUNTERED AFTER COPY OF FILE
         0, RECORD  1
```

ENTER INCREMENT, LOAD CASE NO.
OR NODE NO. FOR OUTPUT ---1

| NODE NUMBER | ORIGINAL COORDINATES .....X.... | .....Y.... | .....Z.... | DISPLACEMENTS .....U.... | .....V.... | .....W.... |
|---|---|---|---|---|---|---|
| 1 | 0. | 0. | 0. | 0.000000 | 0.000000 | 0.000000 |
| 5 | 0. | .600E+01 | 0. | 0.000000 | 0.000000 | 0.000000 |
| 9 | 0. | .120E+02 | 0. | 0.000000 | 0.000000 | 0.000000 |
| 13 | 0. | .180E+02 | 0. | 0.000000 | 0.000000 | 0.000000 |
| 17 | 0. | .240E+02 | 0. | 0.000000 | 0.000000 | 0.000000 |
| 21 | .500E+01 | 0. | 0. | .002320 | .000528 | .016400 |
| 25 | .500E+01 | .600E+01 | 0. | .002550 | .000021 | .019700 |
| 29 | .500E+01 | .120E+02 | 0. | .002210 | -.000000 | .019400 |
| 33 | .500E+01 | .180E+02 | 0. | .002550 | -.000021 | .019700 |
| 37 | .500E+01 | .240E+02 | 0. | .002320 | -.000528 | .016400 |
| 41 | .100E+02 | 0. | 0. | .003450 | .000198 | .056900 |
| 45 | .100E+02 | .600E+01 | 0. | .003710 | .000033 | .058700 |
| 49 | .100E+02 | .120E+02 | 0. | .003080 | -.000000 | .059000 |
| 53 | .100E+02 | .180E+02 | 0. | .003710 | -.000033 | .058700 |
| 57 | .100E+02 | .240E+02 | 0. | .003450 | -.000198 | .056900 |
| 61 | .150E+02 | 0. | 0. | .003570 | -.000147 | .105000 |
| 65 | .150E+02 | .600E+01 | 0. | .003790 | -.000051 | .104000 |
| 69 | .150E+02 | .120E+02 | 0. | .003170 | -.000000 | .103000 |
| 73 | .150E+02 | .180E+02 | 0. | .003790 | .000051 | .104000 |
| 77 | .150E+02 | .240E+02 | 0. | .003570 | .000147 | .105000 |
| 81 | .200E+02 | 0. | 0. | .002830 | -.000477 | .149000 |
| 85 | .200E+02 | .600E+01 | 0. | .002980 | -.000173 | .144000 |
| 89 | .200E+02 | .120E+02 | 0. | .002500 | -.000000 | .143000 |
| 93 | .200E+02 | .180E+02 | 0. | .002980 | .000173 | .144000 |
| 97 | .200E+02 | .240E+02 | 0. | .002830 | .000477 | .149000 |
| 101 | .250E+02 | 0. | 0. | .001560 | -.000696 | .178000 |
| 105 | .250E+02 | .600E+01 | 0. | .001640 | -.000260 | .171000 |
| 109 | .250E+02 | .120E+02 | 0. | .001380 | -.000000 | .169000 |
| 113 | .250E+02 | .180E+02 | 0. | .001640 | .000260 | .171000 |
| 117 | .250E+02 | .240E+02 | 0. | .001560 | .000696 | .178000 |
| 121 | .300E+02 | 0. | 0. | 0.000000 | -.000771 | .189000 |
| 125 | .300E+02 | .600E+01 | 0. | 0.000000 | -.000293 | .181000 |
| 129 | .300E+02 | .120E+02 | 0. | 0.000000 | -.000000 | .179000 |
| 133 | .300E+02 | .180E+02 | 0. | 0.000000 | .000293 | .181000 |
| 137 | .300E+02 | .240E+02 | 0. | 0.000000 | .000771 | .189000 |

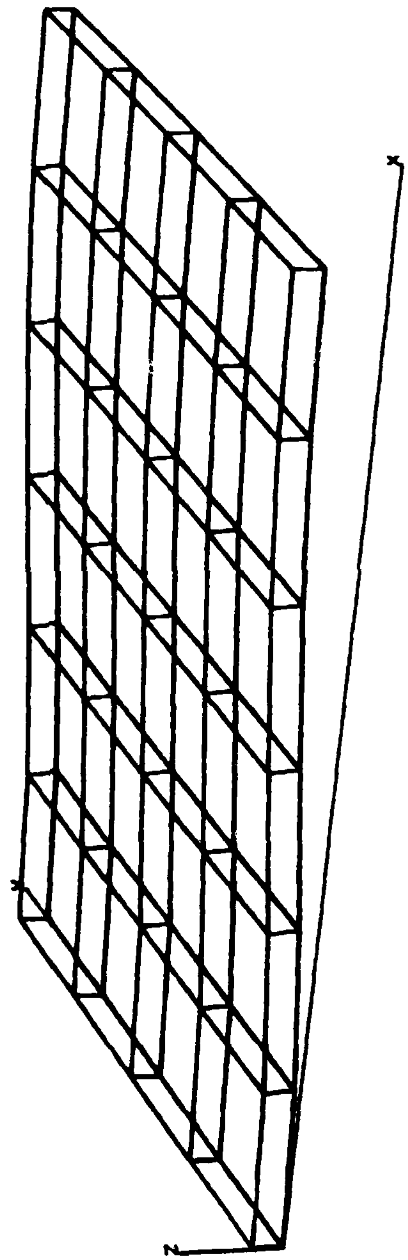
OUTPUT OPTIONS FOR ANALYSIS DATA -

0 - STOP
1 - LIST LOWER SURFACE DISPL.
2 - LIST UPPER SURFACE DISPL.
3 - LIST UPPER FACE STRESSES
4 - LIST LOWER FACE STRESSES
5 - LIST UPPER FACE STRAINS
6 - LIST LOWER FACE STRAINS
7 - LIST CORE LAYER STRESSES
8 - LIST CORE LAYER STRAINS
9 - LIST TIME/FREQUENCY VALS.

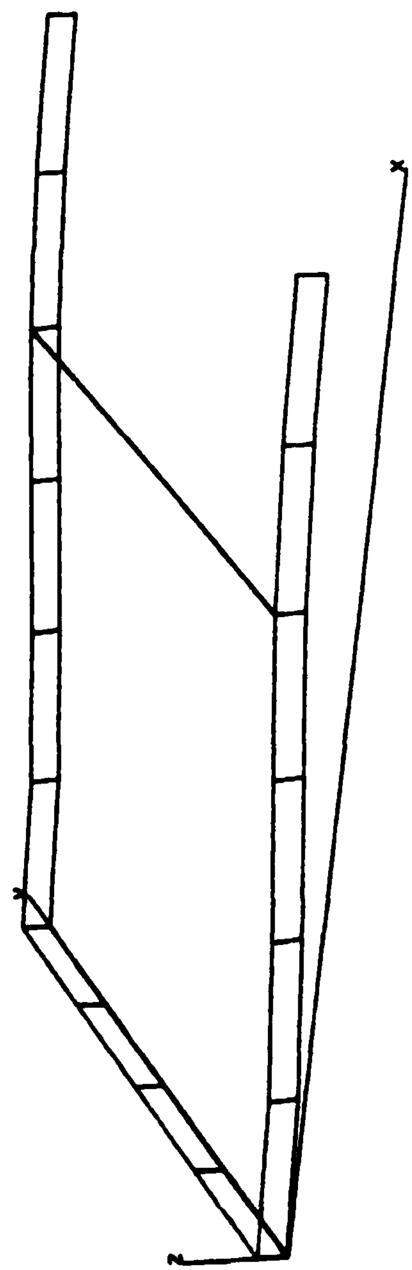
ENTER OUTPUT OPTION NUMBER....7

```
ENTER INCR. OR LOAD CASE NO. -1

STRESS RESULTS FOR CORE LAYER

ELEMENT .....XZ.... .....YZ....
   1       .2180E+02  -.3640E+01
   2       .2110E+02  -.2720E+01
   3       .1660E+02   .6720E+00
   4       .1340E+02   .9300E+00
   5       .7040E+01   .1470E+01
   6       .3320E+01   .1610E+01
   7       .3210E+02  -.2600E+01
   8       .2500E+02  -.2610E+01
   9       .1500E+02   .1120E+00
  10       .1230E+02   .2620E+00
  11       .5190E+01   .7060E+00
  12       .3270E+01   .7770E+00
  13       .3210E+02   .2600E+01
  14       .2500E+02   .2610E+01
  15       .1500E+02  -.1120E+00
  16       .1230E+02  -.2620E+00
  17       .5190E+01  -.7060E+00
  18       .3270E+01  -.7770E+00
  19       .2180E+02   .3640E+01
  20       .2110E+02   .2720E+01
  21       .1660E+02  -.6720E+00
  22       .1340E+02  -.9300E+00
  23       .7040E+01  -.1470E+01
  24       .3320E+01  -.1610E+01

OUTPUT OPTIONS FOR ANALYSIS DATA  -

 0 - STOP
 1 - LIST LOWER SURFACE DISPL.
 2 - LIST UPPER SURFACE DISPL.
 3 - LIST UPPER FACE STRESSES
 4 - LIST LOWER FACE STRESSES
 5 - LIST UPPER FACE STRAINS
 6 - LIST LOWER FACE STRAINS
 7 - LIST CORE LAYER STRESSES
 8 - LIST CORE LAYER STRAINS
 9 - LIST TIME/FREQUENCY VALS.

ENTER OUTPUT OPTION NUMBER....0
  ENTER THE CHARACTERS PER SECOND .............:
    STOP
     23500 MAXIMUM EXECUTION FL.
      .460 CP SECONDS EXECUTION TIME.
EOI ENCOUNTERED AFTER COPY OF FILE
        0, RECORD  1                              120
  TEKTRONIX OR HP PLOTTER(T,H) ................: T
  MODEL INPUT FILE TYPES:
  1 - INPUT DATA FILE
  2 - POSTPROCESSOR (MPOST) FILE
  ENTER FILE TYPE(1,2) ............. ..........: 2
```

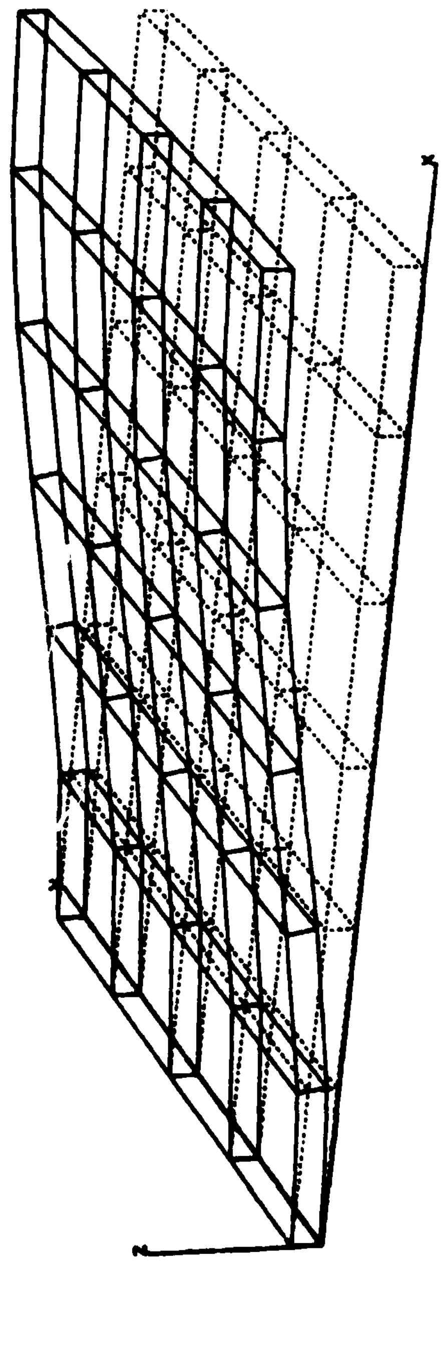
```
 ENTER THE STRUCTURE NUMBER ...................: 1
$HELP
 COMMAND         DESCRIPTION
 $$$$$$$         $$$$$$$$$$$
  AXES            AXES DRAW AND LABEL
  CLIP            CLIP PLANE POSITION
  CUBE            SET MINIMA AND MAXIMA
  DEFAULT         SET DEFAULT VALUES
  DEFORM          DEFORMED MODEL REPRESENTATION
  DRAW            DRAW MODEL
  ELEMENTS        PLOTS ALL OR SELECTED ELEMENTS
  EYE             EYE POSITION
  HELP            LIST ALL COMMANDS
  LABELS          LABELS ELEMENTS AND/OR NODES
  NEW             NEW STRUCTURE
  PROJECTION      PROJECTION TYPE
  REFLECT         REFLECT A PLANE
  ROTATE          ROTATE MODEL ABOUT AXES
  SCALE           SCALE PLOT
  SHRINK          SHRINK ELEMENTS
  STOP            END PROGRAM
  SUMMARY         LIST ALL PARAMETER VALUES
  TIME            PRINT CPU TIME SINCE START OF SESSION
  TRANSLATE       TRANSLATE MODEL FROM ORIGIN
  VERTICAL        VERTICAL AXIS
  ZOOM            ZOOM ON THE MODEL
  SITE            SITE POSITION
$AXES
 PLOT AND LABEL THE AXES?(Y,N) ...............: Y
$EYE
 ENTER THE EYE POSITION,
 XEVE, YEVE, ZEVE ............................: 100,-200,75
$DEFORM
 STRUCTURE PLOT OPTIONS,
 OPTION 1:  UNDEFORMED,
 OPTION 2:  DEFORMED,
 OPTION 3:  BOTH 1 AND 2,
 ENTER THE PLOT OPTION(1,2,3) ................: 2
 UNDEFORMED MINIMA AND MAXIMA
 MINIMUM X    0.            MAXIMUM X    3.00000E+01
 MINIMUM Y    0.            MAXIMUM Y    2.40000E+01
 MINIMUM Z    0.            MAXIMUM Z    9.60000E-01
 ENTER THE TIME/LOAD INCREMENT ...............: 1
 DEFORMED MINIMA AND MAXIMA
 MINIMUM X    0.            MAXIMUM X    3.00000E+01
 MINIMUM Y   -7.71000E-04   MAXIMUM Y    2.40000E+01
 MINIMUM Z    0.            MAXIMUM Z    1.14900E+00
 MAXIMUM DISPLACEMENT OCCURS AT NODE  124
 ENTER THE SCALE FACTOR ......................: 10
$DRAW
```



```
$ELEM
  PLOT ALL THE ELEMENTS?(Y,N) ..................: N
  ENTER THE NUMBER OF ELEMENT TYPES TO PLOT ...: 1
  ELEMENT INPUT OPTIONS
  OPTION 1:   RANDOM ELEMENTS(4,13,64,....)
              WHERE FIRST DIGIT = NO. ELEMENTS
  OPTION 2:   RANGE OF ELEMENTS(1,9,2 FOR 1-9,INC OF 2)
  OPTION 3:   PLOT ALL ELEMENTS
  ENTER THE ELEMENT TYPE .......................: 3
  ENTER THE INPUT OPTION(1,2,3) ................: 3
$DRAW
```



```
$ELEM
  PLOT ALL THE ELEMENTS?(Y,N) ....................! N
  ENTER THE NUMBER OF ELEMENT TYPES TO PLOT ...! 1
  ELEMENT INPUT OPTIONS
  OPTION 1:    RANDOM ELEMENTS(4,13,84,....)
               WHERE FIRST DIGIT = NO. ELEMENTS
  OPTION 2:    RANGE OF ELEMENTS(1,9,2 FOR 1-9,INC OF 2)
  OPTION 3:    PLOT ALL ELEMENTS
  ENTER THE ELEMENT TYPE ..........................! 2
  ENTER THE INPUT OPTION(1,2,3) ...................! 3
$DEFO
  STRUCTURE PLOT OPTIONS,
  OPTION 1:  UNDEFORMED,
  OPTION 2:  DEFORMED,
  OPTION 3:  BOTH 1 AND 2,
  ENTER THE PLOT OPTION(1,2,3) ....................! 3
  UNDEFORMED MINIMA AND MAXIMA
  MINIMUM X    0.             MAXIMUM X    3.00000E+01
  MINIMUM Y    0.             MAXIMUM Y    2.40000E+01
  MINIMUM Z    0.             MAXIMUM Z    9.60000E-01
  ENTER THE TIME/LOAD INCREMENT ...................! 1
  DEFORMED MINIMA AND MAXIMA
  MINIMUM X    0.             MAXIMUM X    3.00000E+01
  MINIMUM Y    -7.71000E-04   MAXIMUM Y    2.40000E+01
  MINIMUM Z    0.             MAXIMUM Z    1.14900E+00
  MAXIMUM DISPLACEMENT OCCURS AT NODE  124
  ENTER THE SCALE FACTOR ..........................! 20
$DRAW
```



```
$STOP
```

```
    047800 MAXIMUM EXECUTION FL.
     3.006 CP SECONDS EXECUTION TIME.
MS   30838  WORDS  (  237180 MAX USED)
CPA     21.717 SEC.          17.898 ADJ.
IO     384.864 SEC.          84.316 ADJ.
CM    5889.456 KWS.          27.460 ADJ.
CRUS                        129.475
COST                  $       8.53
PP     110.271 SEC.      DATE 04/07/81
COMMAND- FILES
--LOCAL FILES--
  ALLOUT     POSTFL   $PROCFIL  $INPUT    $OUTPUT
  INPBAT
COMMAND- BATCH,ALLOUT,PRINT,USER
COMMAND- RETURN,INPBAT,POSTFL,PROCFIL.
```

```
STOP
```

## REFERENCES


1. R. A. Brockman, "Nonlinear Finite Element Analysis of Sandwich Composites," AFWAL-TR-81-3008, Wright-Patterson Air Force Base, Dayton, Ohio, April 1981.

2. F. K. Bogner and M. L. Soni, "Minimum Weight Design of Flat Sandwich Panels Under Edgewise Compression Loads," AFWAL-TR-81-3009, Wright-Patterson Air Force Base, Dayton, Ohio, April 1981.

3. R. A. Brockman, "Stability of Flat, Simply Supported, Rectangular Sandwich Panels Subjected to Combined Inplane Loadings," AFFDL-TR-76-14, Wright-Patterson Air Force Base, Dayton, Ohio, September 1976.

4. R. A. Brockman, "MAGNA Computer Program User's Manual," UDR-TR-80-107, University of Dayton Research Institute, Dayton, Ohio, November 1980.

5. R. A. Brockman and F. K. Bogner, "An Efficient Three-Dimensional Shell Finite Element for Highly Nonlinear Problems," in Advances in Computer Technology - 1980, Volume 2, A. Seireg, editor, A.S.M.E., 1980, pp. 219-226.

6. R. A. Brockman, "A Penalty Function Approach for the Nonlinear Finite Element Analysis of Thin Shells," Ph.D. Dissertation, Department of Mechanical Engineering, University of Dayton, Dayton, Ohio, July 1979.

7. O. C. Zienkiewicz, The Finite Element Method, McGraw-Hill Company, London, 1977.

8. R. B. Coor and A. Jennings, "A Simultaneous Iteration Algorithm for Symmetric Eigenvalue Problems," International Journal for Numerical Methods on Engineering, Vol. 10, pp. 647-663, 1976.

9. R. A. Brockman, M. E. Wright, M. P. Bouchard, and M. J. Hecht, "Structural Analysis Postprocessing Utilities," AFWAL-TR-80-3151, Wright-Patterson Air Force Base, Dayton, Ohio, November 1980.

10. G. R. Monforton, "Discrete Element, Finite Displacement Analysis of Anisotropic Sandwich Shells," Report No. 39, Division of Solid Mechanics, Structures and Mechanical Design, Case Western Reserve University, 1970.

11. H. P. Kan and J. C. Huang, "Large Deflections of Rectangular Sandwich Plates," AIAA Journal, Vol. 5, pp. 1706-1708, 1967.

12. F. J. Plantema, Sandwich Construction, John Wiley and Sons, New York, 1966.

8